Eng. &Tech.Journal, Vol.34,Part (A), No.5,2016

Experimental and Numerical Study on the Improvement of Uniformity Flow in a Parallel Flow Channel

Dr. Wissam H. Alawee
Center of Training, University of Technology/Baghdad
Email;Wissam_772005@yahoo.com
Dr. Jafar M. Hassan
Mechanical Engineering Department, University of Technology/Baghdad
Email: Jafarmehdi1951@yahoo.com
Dr. Wahid S. Mohammad
Mechanical Engineering Department, University of Technology/Baghdad
Email:Wahid_1953@Yahoo.com

Received on:24/6/2015 & Accepted on:17/12/2015

ABSRTACT

The flow uniformity among individual branches within a manifold system can be a significant factor to predict the performance and efficiency of various engineering applications. The non-uniform flow within the manifold system occurs due to the development of vena-contracta at the inlet to the tubes. In order to reduce the flow mal-distribution in the header-lateral system, an approach is presented in this paper. In this approach, the inlet to the lateral pipe is arranged such that the flow enters the lateral pipe with a velocity vector inclined at a certain angle with respect to the cross section of the manifold. Two test sections representing two header structures were used in this study. The first test section is conventional manifold, the second is a modified manifold (the lateral pipes of manifold were tilted a certain angle with axis of manifold). In both test sections, the diameter of the main pipe was 101.6 mm and of the lateral pipe was 50.8 mm. The results of this study showed that the method of changing the angle of water entry to lateral pipes in order to obtain uniform flow has improved the flow distribution by 54% at entry angle of 6^0, that is a reduction in absolute stander deviation from (0.481) to (0.136) and the maximum ratio between highest and lowest flow is 26%. Also, the change in the total flow rate has a slight effect on flow uniformity.

INTROUDUCTION

Flow in Manifolds is employed in wide industrial applications such as water distribution systems, solar collectors, automobile engines and heat exchangers. The design of the manifold can be traced back to the 1950s [1]. For the past several decades, and to a lesser extent up to the present time, 1D model has been used extensively in the manifold design [2–6]. A one-dimensional approach requires many simplifying assumptions. Recently, a numerical simulation has been used for the design of manifolds. References [7-11] are samples of the literature. Manifolds are classified into two categories. One of these categories is dividing manifolds, where there is a one inlet and multiple outlets. The other category is combing manifolds, where there are multiple inlets and a one exit. The main goal of manifold design is to achieve an equal flow rate at each outlet of the distribution manifold.

A major pioneering work has been reported by Burt et al. [12] for pipe spargers. Authors have found that the non-uniformity depends upon the net pressure profile as a result of (a) frictional pressure drop; and (b) pressure recovery due to the reduction in velocity in the direction of flow as the fluid escapes out of the perforations. Further, they have reported that a larger cross sectional area of dividing flow manifolds gives a better flow distribution. Pan et al. [13] performed a three-dimensional computational fluid dynamics (CFD) model to calculate the velocity distribution among multiple parallel channels with triangle manifolds. The effect of channel width and channel spacing on flow distribution among channels with rectangular manifolds has been investigated by Mathew et al [14]. Andrew and Sparrow [15] presented a method to investigate the effect of geometric shape of the exit ports on mass flow rate uniformity effusing from a distribution manifold; three candidate exit-port geometries were considered: (a) an array of discrete slots, (b) an array of discrete circular apertures, and (c) a single continuous longitudinal rectangular slot. Dharaiya et al. [16] studied numerically the effect of tapered manifold shape to increase flow uniformity in mini and micro channels. Tong et al. [17] applied a logic-based systematic method of designing manifold systems to achieve flow rate uniformity among the channels that interconnect a distribution manifold and a collection manifold. The method was based on tailoring the flow resistance of the individual channels to achieve equal pressure drops for all the channels. The tailoring of the flow resistance was accomplished by the use of gate-valve-like obstructions.

The literature survey indicates that flow uniformity is gaining importance in many engineering applications. Also, the flow distribution depends upon header diameter, gravity, header shape, number and tube geometry, etc. Therefore, there is no general method to achieve this goal. The objective of the present experimental and numerical investigations is to modify design for header that can provide significant relief of the mal distribution.

Methodology

Experimental Apparatus

Test Rig

The test rig of this study is shown in **Fig.1**. The rig was assembled at a selected site in fluid laboratory of Mechanical Engineering Department, University of Technology, Iraq. The experimental setup, shown in Figure consists of these parts: the main supply pipe, a test section, a shallow tank to collection water, flow meter, manometer and a centrifugal pump to recycle water to main supply pipe. The water mass flow rate in the test section is controlled by a regulation valve and is measured by a target mass flow-meter. The water flow rate from each lateral pipe was collected in a shallow tank, with dimension (1500x1500x400) mm, and then discharged continuously through pipe diameter 152.4 mm (6 in) to recycle water by centrifugal pump to main supply tank. The water flow rate is measured by a five glass containers with capacity of 50 liter of each container. The containers are placed on a movable support, which allows it to move freely at the same time during experiments. The containers and support are shown in **Fig.2**. Nine pressure tapes are located along the length of the test section. These pressure tapes are used to measure the pressure evolution in different positions in the header and at the inlet manifold.

Figure(1). Experimental test rig

Figure (2): Containers to measure water from outlets

Test Sections

The test section consists of a horizontal header and five parallel lateral pipes. The header is made of acrylic material to ensure the good visibility of developed flow. It is 15000-mm long and its diameter is 101.6-mm. This section is utilized to determine the magnitude of mal-distribution in flow which usually takes place in a conventional manifold. The header inlet is connected to a 3500-mm pipe made of clear polyvinyl chloride (PVC) of 101.6 mm diameter. The long pipe provides a fully developed flow at the header inlet. The branches are regularly 300 mm spaced along the manifold.

According to Jafer 2008 [20] and Wissam 2015 [21], the conventional design of distribution manifold with different applications does not give a uniform flow. It may be expected that in the manifold pictured in **Fig.1** there will be a non-uniform flow distribution such that the smallest mass flow rate will occur in the branch closest to the inlet and the highest flow rate will be encountered in the branch farthest from the inlet. To address this problem, the lateral pipes of manifold were tilted at a certain angle with axis of manifold. This manifold was made of PVC. The header diameter was 101.6 mm and lateral diameter was 50.8 mm. **Fig.3** shows the modified test section used in this study.

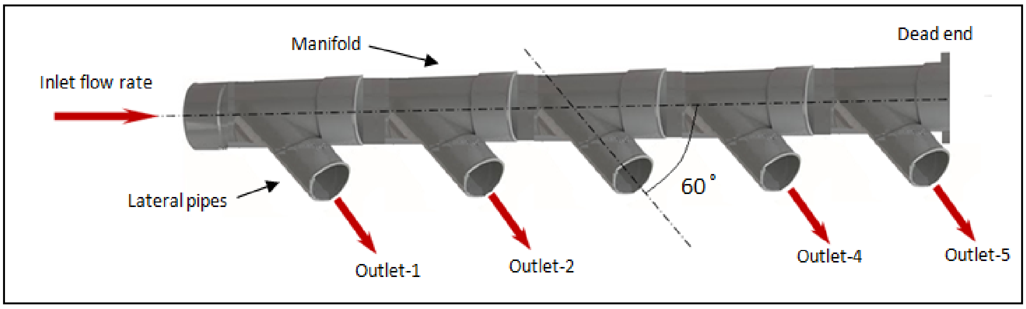


Figure (3): The modified section, the lateral pipes are tilted with axis

Problem Formulation and Numerical Simulation

In the CFD analysis, a model of the distribution manifold has been used. At first, the model of the conventional geometry is used for this study, as shown in **Fig.1**. Then, the simulation was performed to investigate whether the angle of water exit from lateral pipe is a factor in creating uniform outflow. To find the best angle of water exit from the lateral pipe, numerical simulation was made for different cases. The best case has been selected and experimentally tested to make sure of the validity of the numerical model. The modified manifold is illustrated in **Fig.4**. The range of the variation of the angle is listed in **Table 1**.

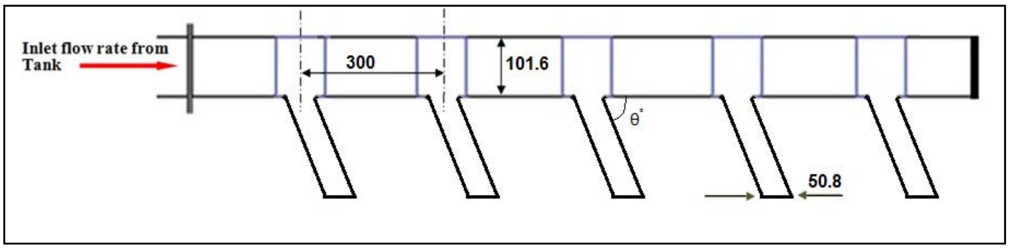


Figure (4): The modify manifold (lateral pipe with different tilted

Table (1): The dimensions of the cases used in this study

case	1	2	3	4
θ°	90	80	70	60

In the present problem, the fluid flow is three-dimensional; that is, all three possible velocity components (x, y, and z) exist, and all three components depend on the three coordinates of Cartesian geometry. The statement of the governing equations for the fluid flow being considered here amounts to writing a set of four partial differential equations: Continuity, x-momentum, y-momentum, and z- momentum, respectively.

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \qquad \ldots(1)$$

$$P\left[\frac{\partial}{\partial x}(u^2)+\frac{\partial}{\partial y}(uv)+\frac{\partial}{\partial z}(uw)\right]=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial u}{\partial z}\right) \qquad \ldots \quad (2)$$

$$P\left[\frac{\partial}{\partial x}(vu)+\frac{\partial}{\partial y}(v^2)+\frac{\partial}{\partial z}(vw)\right]=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial v}{\partial z}\right) \qquad \ldots(3)$$

$$P\left[\frac{\partial}{\partial x}(wu)+\frac{\partial}{\partial y}(wv)+\frac{\partial}{\partial z}(w^2)\right]=-\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial w}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial w}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial w}{\partial z}\right)\ldots(4)$$

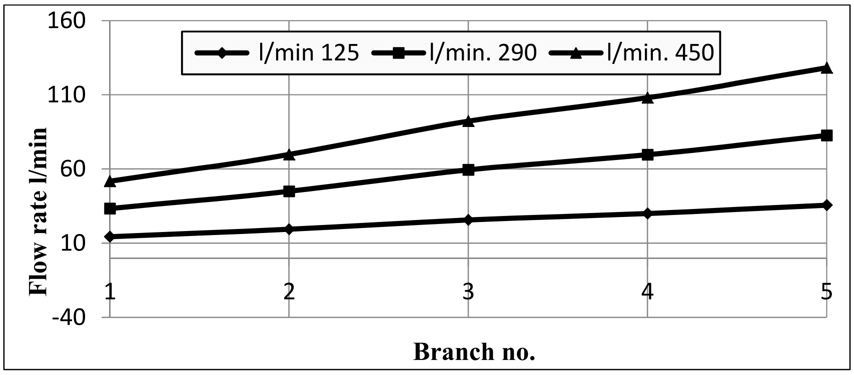
Where: u, v, and w are the velocity components in three dimensions, ρ is the fluid density and μ_{eff}, is the effective viscosity. According to Chen and Sparrow, 2009][18], the turbulence model appropriate for the study of flow in a manifold is the realizable k-ε. Therefore, this model was used here.

Boundary condition Details of the Numerical Algorithm

The first tests were carried out with the reference geometry (a multiple-outlet pipes with dead end) to test the effect of the inlet flow rate on the flow distribution. Inlet flow rates ranges are 100–450 l/min. All tests were performed at room temperature and atmosphere pressure. The simulation of two geometries was done using a commercial CFD software FLUENT. The meshing and boundary definition of the geometries were done using the software, GAMBIT. Tet/Hybrid T-grid scheme was used for the mesh generation [19]. The grid elements in each geometrical model were approximately 2,000,000 elements. When the residuals of the momentum equations are less than 10^{-5}, the solutions are regarded converged.

Results and Discussion

The results of the flow distribution among branches for conventional manifold are given in **Fig.5**.

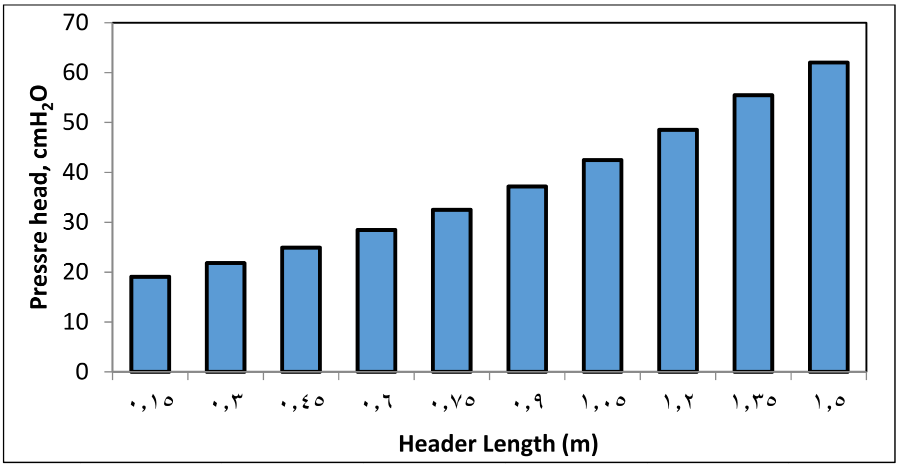


Figure(5): Flow distribution for conventional manifold at different inlet flow rate

As seen in the figure, the water flow in the outlets tends to increase, starting with the first outlet which is badly fed to the last one which is highly fed (more than twice the mean liquid flow rate). This is in agreement with the findings in reference [21][22].

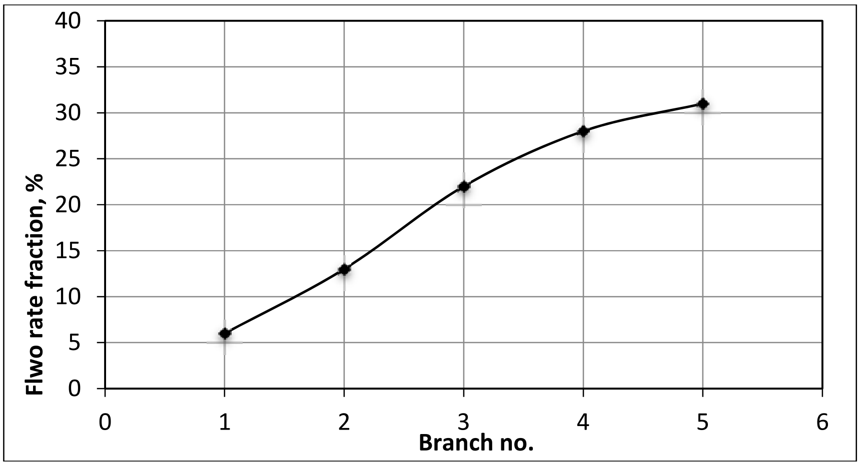
The fact that there is a clear mal-distribution in flow can be explained as follows: There are two factors affecting the flow distribution in manifold: momentum and friction. These factors work in opposite directions to each other. The momentum effect increases the flow along the header whereas the friction effect lowers the flow along the header.

The flow behavior along manifold distribution is in line with the pressure distribution, as shown in **Fig.6**.This figure shows that the pressure increases with increasing downstream distances. Since it is that pressure difference which drives the per-branch flow rate, it is necessary that flow rate also increases with downstream distance.



nifold

The flow ratios (defined as the ratio between of the mass flow rate in the specified branch to the total mass rate [23]) at three different value of inlet flow rates are given in **Fig.7**. The results show, about 6% of total flow is discharged from first branch and about 32% of total flow is discharged from last branch. Also, the **Fig.7** shows the influence of the inlet total flow rate on flow distribution was found to have a slight effect. So, it can be said that the flow distribution of a multi-outlets pipe is independent upon the inlet flow rate within the range (625 l/min. - 950 l/min.).



ld

The main reason for modifying a manifold shape is the expectation that in conventional manifold, the water flow direction by an right angle that leads to formation vortices on The left side of each lateral pipe and the vortices size reduced gradually towards the dead end of the manifold. The fluid flow from each branch depends on these vortices size. **Fig.8** presents the results of flow rate corresponding to the departure angles of 90°. From this figure, it can be seen that a variation angle of inclination for lateral pipe helps to a great extent to improve the flow distribution regardless of inlet total flow rate (the range used in the experiment that is from 625 l/min. to 970 l/min.). **Fig.9** shows the pressure distribution for manifold with inclination lateral pipe. From this figure, the pressure along the manifold was found to be nearly uniform which resulted in a better flow distribution among the branches. **Fig.10** shows the percentage each branch takes from the total flow for manifold with inclined lateral pipe. Comparing these results with those of conventional header, a clear improvement can be seen in flow distribution. For example, when the conventional header is used, the discharge from last branch was about 32% of the total flow and about 6% of total flow is discharged from first branch. This means that the flow exit from first branch is 60% less than that from last branch. For the manifold with inclined lateral pipe, the percentage is reduced from 32% to 21% and the ratio between highest and lowest flow is reduced from 60% to 30%. **Fig.11** presents important characterizing results. These include the standard deviation (STD). The Percentage absolute mean deviation of manifold are 0.481, 0.30, 0.213, and 0.136 at departure angles of fluid of 90°, 80°, 70°, and 60°, respectively. The STD values are lower than that of the typical header with corresponding value of 0.136, as shown in the figure.

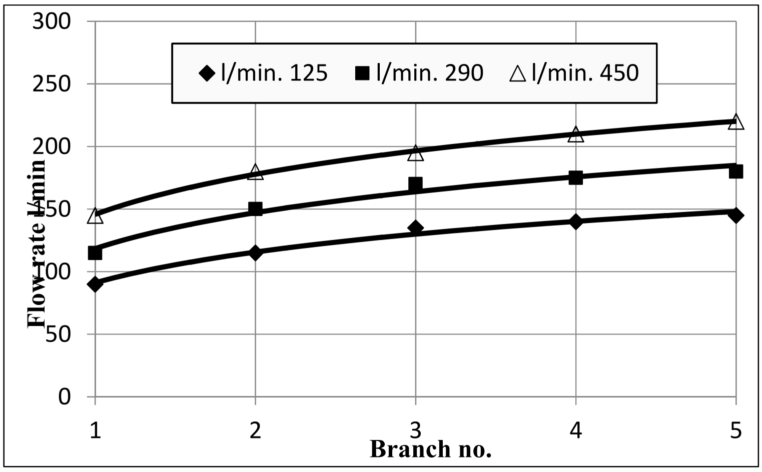
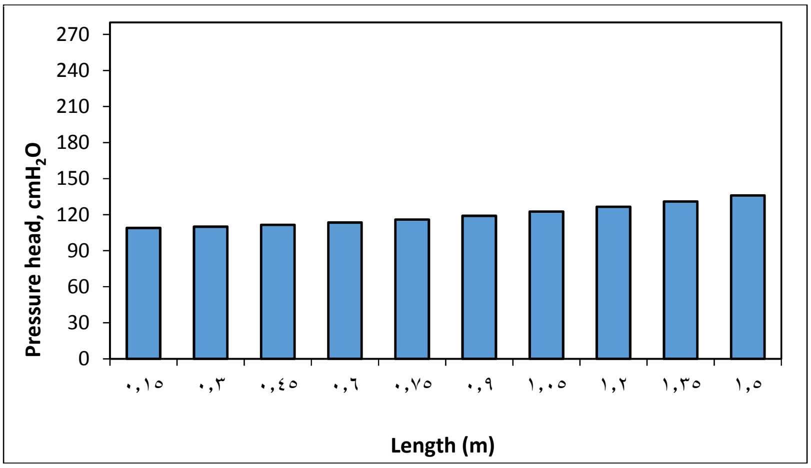
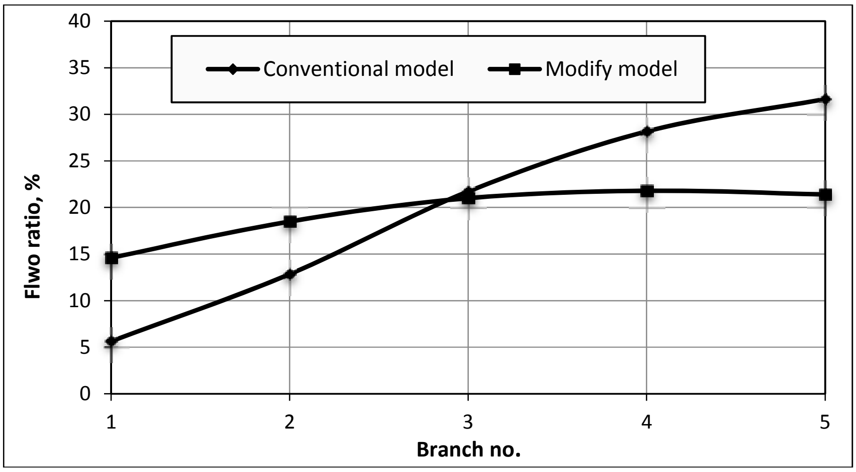


Figure (8): Flow distribution plot for the modify manifold



Figure(9): Variations the manifold pressure head for modify header



old

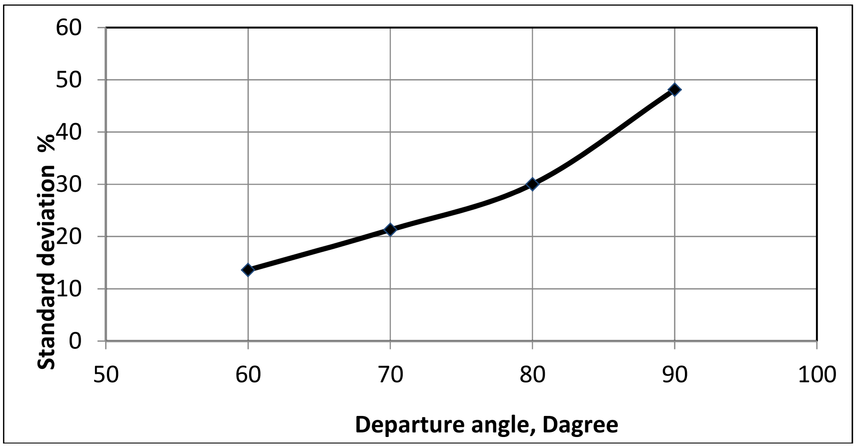


Figure (11): Effect of the modify design on the on flow rate uniformity

Validation of Numerical Results

To validate the numerical simulation of water distribution, a series of experimental tests were carried, showing that the numerical prediction was in a good agreement with the experimental results. The computed and experimental flow rate distribution per-outlet for Qinlet = 790 liter/min. are shown in figs. 11 and 12, respectively. It can be clearly seen from the figures that the difference of flow rate between computed and experimental measured is acceptable (less than 9%).

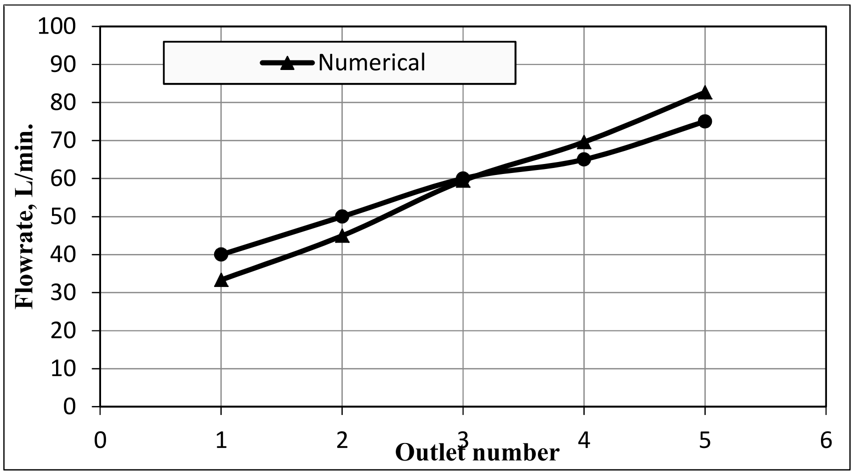


Figure (12): flow distribution per outlet for manifold with conventional model

CONCLUSION

Two test sections representing two header structures were used in this study. The first test section is conventional manifold (the water flow direction by a right angle), the second was a modified manifold (the lateral pipes of manifold were tilted by a certain angle with axis of manifold). In both test sections, the diameter of the main pipe was 101.6 mm and of the lateral pipe 50.8 mm. The results of this study showed that the method of changing the angle of water entry to lateral pipes in order to obtain uniform flow improved the flow distribution by 54% at entry angle of 60°, that is a reduction in absolute standard deviation from 0.481 to 0.136 and the maximum ratio between highest and lowest flow is 26%. Also, that change in the total flow rate has a slight effect on flow uniformity. Therefore, it can be safely said that the inlet flow rate has no effect on flow distribution.

Acknowledgements

The authors would like to thank the Mechanical Engineering Department, University of Technology for its assistance.

REFERENCE

[1].Tong J.C.K., Sparrow E.M., Abraham J.P., Attainment of flow rate uniformity in the channels that link a distribution manifold to a collection manifold, J. Fluids Eng. 129 (2007) 1186–1192.

[2].Spengos, A. C., and Kaiser, R. B., 1963, “Tapered Manifold Flow Spreader,” Tappi J., 46, pp. 195–200.

[3].Leydorf, Jr., G. F.,, Minty, R. G., and Fingeroot, M., 1972, “Design Refinement of Induction and Exhaust Systems Using Steady-State Flow Bench Techniques,” SAE Paper No. 720214, p. 23.

[4].Bajura, R. A., and Jones, E. H., 1976, “Flow Distribution Manifolds,” ASME J. Fluids Eng., 98, pp. 654–666. Furzer, I. A., 1980, “Steady State Flow Distributions in a Plate Column Fitted with a Manifold,” Chem. Eng. Sci., 35, pp. 1291–1298.

[5].Hsu, C., 1985, “How to Achieve Balanced Cooling with Internal Manifolding,” Proceedings Annual Technical Conference, Washington, DC, April 29– May 2, Society of Plastics Engineers, pp. 759–760.

[6].Riggs, J. B., 1987, “Development of Algebraic Design Equations for Dividing, Combining, Parallel.

[7].Childs, E. P., 1987, “Flow Maldistribution in Disc-Type Power Transformer Windings,” ASME, Heat Transfer Division, 75, pp. 137–143.

[8].Poh, S. T., and Ng, E. Y. K., 1998, “Heat Transfer and Flow Issues in Manifold Microchannel Heat Sinks: A CFD Approach,” Proceedings of 2nd Electronics Packaging Technology Conference, Singapore, December 8–10, pp. 246–250.

[9].Ng, E. Y. K., and Poh, S. T., 1999, “Investigative Study of Manifold Microchannel Heat Sinks for Electronic Cooling Design,” J. Electron. Manuf., 9, pp. 155–166.

[10].Commenge, J. M., Falk, L., Corriou, J. P., and Matlosz, M., 2002, “Optimal Design for Flow Uniformity in Microchannel Reactors,” AIChE J., 48, pp. 345–358.

[11].Karki, K. C., Radmehr, A., and Patankar, S. V., 2003, “Use of Computational Fluid Dynamics for Calculating Flow Rates Through Perforated Tiles in Raised-Floor Data Centers,” HVAC&R Res., 9, pp. 153–156.

[12].Burt, C. M.,Walker, R.E.,Styles, S. W., 1992, Infiltration System Evaluation Manual. Irrigation Training And Research Center,. California polytechnic state, Univcity San Luis Obispo, CA.

[13].Pan, M., Zeng, D., Tang, Y., & Chen, D. “CFD-based study of velocity distribution among multiple parallel microchannels”, Journal of Computers, 4 (2009) 1133-1138.

[14].Mathew, B., John, T. J., & Hegab, H. “Effect of manifold design on flow distribution in multichanneled microfluidic devices”, In 2009 ASME Fluids Engineering Division Summer Conference, Vol.2 (pp. 543–548).

[15].Andrew W. Chen, Ephraim M. Sparrow, “Effect of Exit-port Geometry on the Performance of a Flow Distribution Manifold”, Applied Thermal Engineering, Vol. (29), pp. (2689-2692), 2009.

[16].V. V. Dharaiya at el. “evaluation of a tapered header configuration to reduce flow maldistribution in minichannels and microchannels” Proceedings of the ASME 2009 7th International Conference on Nanochannels, Microchannels and Mini-channels.

[17].J.C.K. Tong, E.M. Sparrow, J.P. Abraham, Attainment of flowrate uniformity in the channels that link a distribution manifold to a collection manifold, J. Fluids Eng. 129 (2007) 1186–1192.

[18].Chen, A. W., and Sparrow, E. M., Turbulence Modelling for Flow in a Distribution Manifold,” Int. Journal of Heat Mass Transfer, 52 (6) (2009) 1573–1581.

[19].Dharaiya, V. V., and Kandlikar, S. G., 2009, Evaluation Of A Tapered Header Configuration To Reduce Flow Maldistribution In Minichannels And Microchannels, Proceedings of the ASME 2009 7th International Conference, June 22-24, 2009, Pohang, South Korea.

[20].Hassan, J. M.,2008, Flow Distribution in Manifolds, Journal of Engineering and Development, vol. 12, no.4, pp. 159-177.

[21].Alawee, W. H., 2015, Solution for the Fluid Flow Distribution Manifold Problem, Ph.D thesis Department of Mechanical Engineering, University of Technology.

[22].Pertorius W.A., Dividing- flow Manifold Calculations with a Spreadsheet, water science, 23 (2) (1997) 147-150.

[23].Chiou J.P., The effect of nonuniform fluid flow distribution on the thermal performance of solar collector, Journal of Solar Energy, 29 (1982) 487–502.